Defense Technical Information Center
Compilation Part Notice

ADP023648

TITLE: Turbine Burners: Turbulent Combustion of Liquid Fuels

DISTRIBUTION: Approved for public release, distribution unlimited

This paper is part of the following report:

TITLE: Army Research Office and Air Force Office of Scientific Research
Contractors' Meeting in Chemical Propulsion Held in Arlington, Virginia
on June 12-14, 2006

To order the complete compilation report, use: ADA474195

The component part is provided here to allow users access to individually authored sections
of proceedings, annals, symposia, etc. However, the component should be considered within
the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP023616 thru ADP023650

TURBINE BURNERS: TURBULENT COMBUSTION OF LIQUID FUELS

William A. Sirignano, Feng Liu, and Derek Dunn-Rankin

Department of Mechanical and Aerospace Engineering
University of California, Irvine
Irvine, CA 92687-3975

SUMMARY/OVERVIEW:
The proposed theoretical/computational and experimental study addresses the vital two-way coupling between combustion processes and fluid dynamic phenomena associated with schemes for burning liquid fuels in high-speed, accelerating and turning transonic turbulent flows. A major motivation for this type of combustion configuration results from the demonstrated potential for immense improvements in the performance of gas turbine engines via combustion in the turbine passages. This program will address various fundamental issues concerning liquid-fuel combustion in an axially and centrifugally accelerating flow. The major combustion challenge involves ignition and flame-holding of the spray flame in the high-acceleration flow and associated optimization of the injection of the liquid fuel and some secondary air into a protected recirculation zone provided by a cavity.

TECHNICAL DISCUSSION:
This three-year study began on March 16, 2006. So, only limited progress can be reported. The study focuses on the upstream subsonic portion of the accelerating flow where the critical ignition and flame-holding processes occur. This research is substantially different from the studies of liquid fuel injection and burning in high-speed flows motivated by the Scramjet application because we consider injection into the low-speed portion of the flow, where the flow begins its acceleration. The combustion can extend, however, into regions of high acceleration and transonic velocity. These high accelerations in the stratified flow can have profound consequences on the mixing and flame-holding. Liquid fuel will be injected into a recirculation zone (e.g., cavity) adjacent to the accelerating flow in a curved duct. A two-phase, unsteady simulation is being developed for the reacting, multicomponent turbulent flow using Lagrangian coordinates for the droplets and Eulerian coordinates for the gas phase. In this situation, where vaporization can be a controlling process, quasi-steady vaporization models are inadequate and transient droplet behavior must be considered. Experiments will be performed with non-intrusive diagnostics for reacting and cold-flow mixing studies, both to provide inlet and boundary conditions for the computations and to verify numerical simulations.

Experimental Approach: The experimental apparatus is designed to be flexible in its ability to simulate processes that would occur with fuel addition in the curved passages of a turbine burner. Although typical nozzle guide vane geometries can be quite complex, the experiments will concentrate initially on an auxiliary combustion chamber and its interaction with the primary geometric curvature that guides the hot gases into the turbine rotor. The

experiments can also provide global evaluation of flow uniformity, flow stability, pressure losses, pollutant formation, and efficiency.

Figure 1 gives an overview of the experimental apparatus. The key feature of the experimental apparatus is to produce a combusting environment in a flow that has both centrifugal and axial accelerations by curving the duct and decreasing duct cross-sectional area with downstream distance. To simplify the grid behavior in the related numerical simulations, only one of the duct dimensions decreases. The plenum section includes a heater section and flow straightening. Both electrical preheating and a burner heater are included to provide a wide range of operating conditions. The hot-flow curved duct section is primarily ceramic coated mild steel, but it has several windows to provide optical access into reacting spray flow experiments. Liquid or gaseous fuel can be injected into the curved duct through ports upstream or downstream of the auxiliary cavity. The auxiliary segment is the heart of the experiment. It allows a variety of fuel injection locations, geometries, and orientations relative to the curved section. Windows provide full optical access to this section. The inlet channel cross-section is 100 mm x 75 mm, decreasing to 50 mm x 75 mm at the outlet, with a channel radius of curvature between 200 and 300 mm.
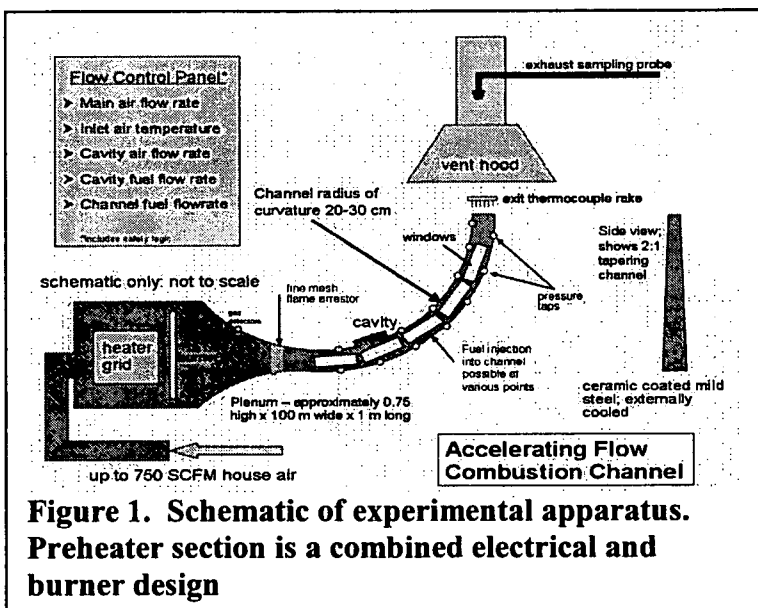

Figure 1. Schematic of experimental apparatus. Preheater section is a combined electrical and burner design

The key to flame-holding, particularly in marginally heated or unheated upstream air flows, is fuel/air mixing and the protection of the ignition kernel from the high-acceleration shearing flows of the main curved duct. The mixing in the curved duct should be rapid enough to establish a combustible mixture when fuel is introduced upstream. The shear rates are high, however, and there is a required mechanism for flame-holding (i.e., a low-velocity, low-stretch-rate region where the flame can be maintained). Therefore, the turbine-burner configuration will use the small auxiliary chamber as a rearward-facing step to produce a relatively low speed vortex into which fuel can be injected. A similar approach has shown promise in some preliminary activities at the Air Force Research Laboratory.

The auxiliary chamber of the hot-flow duct houses the cavity volume and is a removable piece to allow flexibility in optical access, chamber shape, chamber size, and fuel injection location. The cold-flow experiments and the numerical simulations will have provided some guidance with regard to which configurations provide good mixing and flow field characteristics, but reaction and heat release are certain to change some of these findings. Optical access allows detailed observation of the spray and its mixing into the recirculation

zone of the chamber. The cavity will allow fuel injection in at least 6 different locations to determine the interaction between the in-chamber vortex motion and the additional momentum provided by the spray. The hot-flow experiment requires a substantial development and construction effort. We have settled on the dimensions and fabrication strategy of the plenum section and the curved duct section. Heater design, using flexible inconel heater elements, including a verification of electrical power infrastructure is completed. Detailed design of the cavity section is underway.

In cold flow conditions, we will first use acetone PLIF to assess fuel mixing from the auxiliary chamber, along with static pressure measurements along the curved duct walls and pitot pressure surveys. To these measurements we will add particle image velocimetry (PIV) of the flow in the duct under some of the key conditions. We have a PIV system, but will add seeding capability to the experiment. In the hot-flow experiments, it may be difficult to have optical access in the curved walls without leaks and without compromising the curvature. Nevertheless, we will use discrete window segments to observe particular regions of the flow. All windows in the system are flat to reduce the optical complexity of the alignment of overlapping beams. The first experiments will be chemiluminescence and schlieren visualizations of flame location.

Both local and global experiments will be coordinated with the calculations to provide detailed comparison data most critical to the predictions and performance of the process. For example, PIV will give instantaneous velocity vector plots on the centerline of the duct near the auxiliary chamber to show how the flows interact. These same velocities will be predicted by the numerical code. Similarly critical particle size distribution data for injected fuel droplets will serve as inputs to the 2-phase calculations. Consultation with colleagues at the AF Research Laboratories will continue.

Computational Approach: The first approximation to the problem is a two-dimensional rectangular channel flow over a rectangular cavity, as shown in Figure 2. The numerical method used is a finite-volume, unsteady laminar code for viscous, multi-component, reacting flow, with a non-uniform Cartesian mesh. Continuity, momentum, energy, and species equations are solved simultaneously to within a 0.1% residual error. Viscosity, thermal conductivity and specific heat are modeled as functions of temperature. Density is a function of both temperature and pressure, using the assumption of a perfect gas. Binary diffusion coefficients are modeled as functions of temperature and density using kinetic theory approximations. Each species is considered to diffuse with nitrogen, the dominant species. The imposed boundary conditions for the equations are as follows: constant velocity profile at channel inlet, constant pressure at channel outlet, parabolic-velocity profile at injection points, and no-slip condition on all walls; constant inlet temperature, constant wall temperature, constant injection temperature; specified inflow species concentrations and zero normal gradient at walls for species concentration. In addition, Lagrangian derivatives of the outflow are zero.

The first calculations consider non-reacting flow at a Reynolds number of 5000. The three sides of the cavity have been considered as general injection locations for the fuel. In Figure 2, gaseous heptane is injected into the left-side of the cavity, normal to the wall.

The figure shows the concentration of the heptane as it diffuses with the air in the channel. Here, the heptane is injected at 300K at 5 m/s, and the channel inlet flow is at 1000K and 12m/s. The mass flow rate of heptane is approximately 25% of the mass flow rate of air in the channel.

The results to date have shown limited mixing of the injected fuel with the air in the cavity, with the injected fuel forming a boundary layer along the bottom wall of the channel as it exits the cavity. Several extra vortices are created inside the cavity when fuel is injected, depending on the injection location and velocity. An additional vortex also occurs downstream of the cavity on the bottom wall of the cavity for some injection conditions.
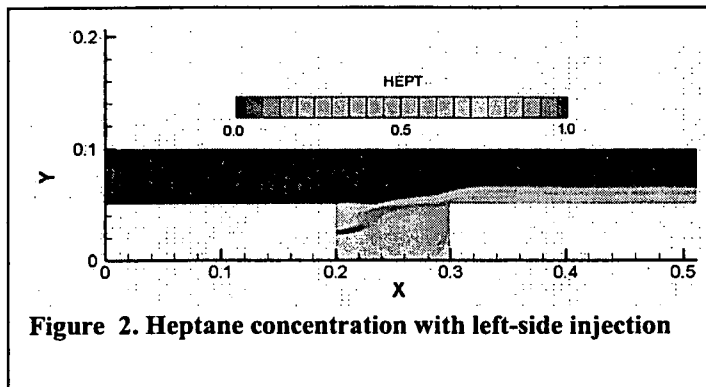

Figure 2. Heptane concentration with left-side injection

The plan is to develop the code to consider multiple points of injection of fuel or fuel with air in the cavity and at points upstream of the cavity. In an incremental way, additions/ modifications to the code will allow for liquid-fuel injection and vaporizing spray analysis, oxidation reaction, three-dimensional effects, and turbulent modeling. Choices of parameters and sub-grid models will be guided by our experimental observations and those at the Air Force Labs, plus our previous and concurrent experience with computations of a reacting mixing layer in an accelerating and turning channel flow.

Previous calculations for reacting gaseous mixing layers indicate that the flow in the channel is transitional rather than fully turbulent. Density variations in the channel lead to interesting effects whereby not only a shear-layer (Kelvin-Helmholtz) instability occurs but the turning can result in Rayleigh-Taylor and centrifugal fluid instabilities. We will look for signs of these effects in the two-phase flow with a cavity. An open question that remains is whether it is superior to model the flow in the interesting Reynolds number range as an unsteady, three-dimensional flow with some sub-grid turbulence modelling or to use some Favre-averaged equations.